Internal Threading: Pilot Hole Considerations

Pilot Hole Diameter Calculations:

A hole must be created prior to making an internal thread. A pilot hole that is too small will result in greater expense caused by damaged tools and extra time expended creating the threads. A pilot hole that is too large will reduce the strength and effectiveness of the threads. The designer needs to understand the interplay and relationship of the variables determining the choice of pilot hole diameters. A pilot hole is made with a *tap drill* when a tap is used to create the internal threads.

Terminology: The Pilot_Dia is the size of the pilot hole allowing internal threads to be made. %_of_Thread is the percentage of full thread form allowed by the pilot hole being used. Maj_Dia is the major diameter of the screw thread under consideration. TPI is the number of threads per inch used to define Unified National threads. Pitch is the measure of the thread's crest-to-crest distance used most commonly with metric threads. So long as units are consistent, TPI = 1/Pitch and Pitch = 1/TPI. %_of_Load is the percentage of full thread load carrying capacity left in an internal thread when a pilot hole creating less than 100% of full thread is used. T_Factor is a numeric value accounting for differences in manufacturing processes.

There are two ways to make internal threads: cutting and forming. Cutting involves the removal of material leaving the threadform. Forming involves moving material to shape the threadform. A formed thread causes the metal's grain to flow along the contour of the threadform and, generally, work-hardens the material – both of which act to create a stronger and tougher thread. Forming a thread requires a rigid set-up, greater torque, care in lubrication, and close control of the pilot hole diameter to be successful. Cutting a thread has more allowance for error. A cut-thread T_Factor is 76.98. A formed-thread T_Factor is 147.1.

Unified National: Pilot_Dia = Maj_Dia – (%_of_Thread / (T_Factor * TPI))

Metric: Pilot_Dia = Maj_Dia – ((%_of_Thread * Pitch) / T_Factor)

While these equations are differentiated as **Unified National** and **Metric**, the equations will work with either so long as TPI = 1/Pitch and units are not mixed. The distinction is made to use definitions where Unified National threads are denoted by Maj_Dia, TPI, and class of fit while Metric threads are denoted by Maj_Dia, Pitch, and class of fit. The equations are set to use common thread definitions. These equations may be solved for %_of_Thread as:

Unified National: %_of_Thread = (Maj_Dia – Pilot_Dia)(T_Factor * TPI)

Metric: %_of_Thread = (Maj_Dia – Pilot_Dia)(T_Factor / Pitch)

The advantage of this format of the equations is that they may be used with a measured value for the pilot hole diameter so long as the units are consistent. Measurement-based calculations always use the cut-thread T_Factor value (76.98). The form-thread T_Factor value (147.1) allows for material that will have moved during the threading process. It is also recommended that you make several measurements of the inside diameter of a thread and average them to create the Pilot_Dia value as metal will often not flow evenly during the forming process. When you have found the %_of_Thread, the %_of_Load value may be calculated as:

$$\%\_of\_Load = 100 * [1 – (1 - \%\_of\_Thread/100)^2]^2$$

The table below shows the effect of %_of_Thread on the %_of_Load value for cut threads. Formed threads will have higher values based on (A) the Poisson's ratio of the material being

 Revision A

threaded and (B) the ability of the material to accept work hardening (also called *strain hardening*). Note that a pilot hole sized for 75% of full thread will reduce the strength of the connection by a mere 12.1%. Using a pilot hole sized for 75% thread will typically double tap life when compared to a pilot hole sized for 100% thread.

% of Thread	% of Load	% of Thread	% of Load	% of Thread	% of Load
100	100.0	80	92.2	60	70.6
95	99.5	75	87.9	55	63.6
90	98.0	70	82.8	50	56.3
85	95.6	65	77.0	45	48.7

Application:

A quick comparison of pilot hole values for a ¼-20 nominal cut-thread gives us:

%_of_Thread:	Pilot_Dia (Calc):	Drill Diameter:	%_of_Load:
100%	.1850	.1850 (#13)	100.0%
90%	.1915	.1910 (#11)	98.0%
75%	.2013	.2010 (#7)	87.9%
60%	.2110	.2130 (#3)	70.6%

A recommended exercise is to tap each of these pilot hole diameters using a ¼-20 tap and compare the torque and speed of operation. This will give you an intimate feel regarding the value of maximizing the pilot hole's size. Another worthwhile exercise is to tap a given pilot hole size using different lubricants to understand their effect on the process.

Taking data from the **Unified National Thread Data** charts (see TBD), the Nut Shear Area per unit length of engagement values for a ¼-20 nominal thread are:

Thread Size & Class:	Nut Shear Area/L:
.2500-20UNC-1B	.4740 in^2/in
.2500-20UNC-2B	.5398 in^2/in
.2500-20UNC-3B	.5625 in^2/in

Thus, if we had a 1.000 inch long engagement of a .2500-20UNC-1B hole with 100% of full threads, we would have .4740 in^2 of shear area carrying the load. If the material into which these threads were created had a minimum shear strength of 28,000 psi (i.e. ASTM A-36 steel), then the threads would have a minimum load carrying capacity of (28,000 psi * .4740 in^2/in * 1.000 in =) 13,272 lbs.

That same interface using a 75% pilot hole will have (87.9/100 * 28,000 psi * .4740 in^2/in * 1.000 in =) 11,666 lb minimum load carrying capacity – 87.9% of the value calculated for a 100% thread. If the engagement is reduced to .250 inches, the load carrying capacity would be reduced to 25% of the values cited here.

Warning: Screws have an allowance of up to 3 imperfect threads on their end and screw lengths are toleranced such that they will be shorter than their nominal values. Also, nuts may have up to 3 imperfect threads allowing for the chamfers that make them easier to start. These conditions **must** be taken into account when calculating engagement lengths for load carrying values!

Traditional 60° Sharp Threads:

The information in this article is designed for use with modified crest and root threads as defined in modern thread standards. Traditional (*American Standard*) sharp threads do not fit these definitions. While such threads are still in common use, they do not meet the Unified National, ANSI, ISO, BS, DIN, or JIC threadform specifications and should be noted as being more susceptible to failure by fatigue or stress concentrations. Good mechanical design practice avoids their use. However, when required, information on applying such threads may be found in *Machinery's Handbook*, Shigley's *Standard Handbook of Machine Design*, *The American Machinist's Handbook*, and other sources.

The traditional formula for the Pilot_Dia for a 60° sharp internal thread is:

Pilot_Dia = Maj_Dia - .975/TPI

This will provide an approximate 75% thread.

Pilot Hole Depth Considerations for Cut-Thread Taps:

A blind pilot hole must provide the necessary length for thread engagement, allow for the tap's imperfect lead threads, allow for the point of the tap, provide clearance for chips that are not cleared by the tap during use, and allow "over-twist" to insure that a full threadform is cut in higher shear strength materials. This requires that the designer know what type of tap will be used and what the impact of that choice is on manufacturing cost.

For cut-thread type taps, there are three basic types of taps: taper taps, plug taps, and bottoming taps. A taper tap is allowed to have 7-10 imperfect threads to distribute the cutting force. A plug tap is allowed to have 3-5 imperfect threads to distribute the cutting force. A bottoming tap is allowed to have 1-2 imperfect threads to distribute the cutting force. The general rule is that the greater the number of threads over which the cutting force is spread, the longer the taps will last. The downside is that a deeper pilot hole is required to clear the non-full-thread end of the tap.

The tip of a tap protrudes beyond the flutes no more than the minor diameter of the thread being cut.

As the tap is driven into the pilot hole, some of the chips will be pushed forward of the tap and pose the danger of jamming and breaking the tap. This situation may be avoided by removing the tap and blowing the chips from the hole – at added expense. It is better to allow for this and avoid jamming the tap. If the material has small granular-type chips (such as cast iron), you need to add an additional depth of 12.5% of the major diameter for each major diameter of depth to be threaded. If the material has medium sized, non-stringy chips (such as 12L14), you need to add an additional depth of 25% of the major diameter for each major diameter of depth to be threaded. If the material has long stringy chips (such as merchant steel), you need to add an additional depth of 40% of the major diameter for each major diameter of depth to be threaded. It is best to allow a minimum of 150% of the major diameter depth for clearance whenever such an allowance is possible.

The higher the shear strength of the material being tapped, the more likely it is that some of the material will *reform* when the tap is removed. An additional turn or partial turn of the tap is used to account for this. This becomes more important as the pitch of the thread increases. If the material to be tapped has relatively low shear strength, no additional allowance is required. If the material to be tapped has moderate shear strength, an additional ¼ turn of the tap should be allowed. If the material to be tapped has high shear strength, an additional full turn of the tap should be allowed.

Putting all this together gives us: Type of tap (taper, plug, or bottoming) which determines the imperfect thread clearances required. A taper tap needs to allow for 10 imperfect threads which gives us 11*Pitch as the additional depth of the pilot hole. A plug tap needs to allow for 5 imperfect threads which gives us 6*Pitch as the additional depth of the pilot hole. A bottoming tap needs to allow for 2 imperfect threads which gives us 3*Pitch as the additional depth of the pilot hole. We add to this value the minor diameter of the screwthread to allow for the point of the tap and this gives us the *total tap allowance* required.

Next, we need to figure out the *Major_Depth* of the threads in the hole. This is defined as being the nominal threaded depth divided by the major diameter of the screwthread. This is multiplied times the chip allowance ($^{1}/_{8}$ for small granular chip materials, ¼ for medium chip materials, or $^{2}/_{5}$ for long stringy chip material). Thus, *Major_Depth * chip_allowance_factor* gives us the *total chip allowance* required.

Finally, we find the *shear-reform* allowance value. We start by finding the *shear-reform-factor* For low shear strength materials, this value is 0. For medium shear strength materials, this value is ¼. For high shear strength materials, this value is 1. Multiplying this value by the pitch of the screwthread (*shear-reform-factor**Pitch) gives us the *total shear-reform allowance* required.

The nominal pilot hole depth for a cut-thread tap is the sum of these parts. I.E.:

Nominal_Pilot_Depth = required_thread_depth + total_tap_allowance + total_chip_allowance + total_shear-reform_allowance.

Violation of this rule-set will result in extra expense in handling or a higher probability of tap breakage – which adds additional costs too.

Tips & Tricks When Shallow Pilot Holes Must Be Used: OK, we know that this is, shall we say, *less than optimum practice*, but what do you do when you are forced into a corner and have to tap a hole using a shallow pilot hole? The answer is that you *punt* – but what are the *punts* available to you? They are:

1. Use a form-thread tap. If the material is ductile enough, this is the best possible solution to this problem.
2. Mill the thread. This is a good solution if it can be done.
3. Use a spiral tap set to maximize chip ejection. This usually requires that you start with a plug tap and finish with a bottoming tap.
4. Fill the hole with a wax compound (paraffin and beeswax usually work quite well) such that the wax will extrude back out the hole carrying away the chips.
5. Backfill and plug the hole after tapping. Crude, rude, but this is sometimes the only real solution to the requirement of a shallow pilot hole.

Please, be polite, note on the drawing that you acknowledge that this is a *punt* and let the shop know that you understand that you have thrown a monkey on their back!

Pilot Hole Depth Considerations for Form-Thread Taps:

A form-thread tap requires much less depth clearance in a blind pilot hole than cut-thread taps. Form-thread taps come in plug and bottoming styles. A plug-thread tap has a forming lead of 4 threads. A bottoming-thread tap has a forming lead of 2 threads. The tip of the tap is usually either flat or concave and requires no special clearance. The factor people forget when forming threads in a blind pilot hole is to allow a clearance volume for the lubricant that may be pushed ahead of the form-thread tap. Compressed pockets of lubricant can often blow out the part or fatally jam the tap!

An allowance of 3*Pitch for bottoming taps or 5*Pitch for plug taps provides the *total tap allowance* for form-thread taps.

An allowance of ¼ the major diameter plus $^1/_8$ the major for each major diameter of thread depth is usually sufficient to prevent hydraulic damage. The ¼ major diameter allowance provides clearance for the drops of lubricant that tend to form at the tip of the tap. Careful application of the lubricant and cleaning the tip of the tap **can** allow this factor to be avoided – at extra cost and danger of tap breakage. Using the *Major_Depth* definition above, we can restate this as: *total hydraulic allowance* = ¼ * *Major_Dia* + $^1/_8$ * *Major_Depth*.

This gives us:

Nominal_Pilot_Depth = *required_thread_depth* + *total_tap_allowance* + *total_hydraulic_allowance.*

Conclusion:

The diligent designer understands how their decision affect the price and performance of the product they are creating. This chapter provides the basic rules necessary to optimize the use of taps to create internally threaded holes. Following these rules will help make your designs more cost effective. However, this is only an introductory chapter. Many people have spent their lives developing detailed analysis of screwthread application, cost, and failure modes. This chapter is not intended to provide analysis tools for every application. It is up to the reader to assure themselves of the appropriateness and rigor of these rules to their application.

Internal Threading: Standard Drill Charts

Standard tap drill charts are common. They are based on having a nominal 75% of full thread in the tapped hole. The previous statement is both true and false. The tap drill recommendations in these charts are standard (American) drill bit sizes and do not necessarily provide an actual 75% of full thread. More importantly, drill bits tend to create holes slightly larger than their nominal diameter in most materials. This is something the astute designer will recognize.

The following charts are based on this assumption set. Please note that a *% Thd* value is given as reference to the theoretical percentage of full thread based on the nominal drill size. While the reader should be aware of this value, they should also be aware of the inherent error underlying the analysis.

Caveat: The *Formed Thread Tap Drill Chart* provides **book values**. They are only a general guide. The actual value needed is dependent upon: form tap geometry; the Poisson's ratio of the material being tapped; the Tangent or Secant modulus of the material being tapped; and the type and amount of lubricant used in the process. The best recommendations for preparing holes for formed thread tapping will come from the manufacturer of the tap you intend to use. It should be understood that such information trumps generalized calculations.

Cut Thread Tap Drill Chart

Thread:	Tap Drill:	Tap Drill:	% Thd:
.0600-80UNF	.0469	3/64	80.8%
.0730-64UNC	.0595	#53	66.5%
.0730-72UNF	.0595	#53	74.8%
.0860-56UNC	.0670	#51	81.9%
.0860-64UNF	.0700	#50	78.8%
.0990-48UNC	.0785	#47	75.7%
.0990-56UNF	.0810	#46	77.6%
.1120-40UNC	.0860	#44	80.1%
.1120-48UNF	.0890	#43	85.0%
.1250-40UNC	.0995	#39	78.5%
.1250-44UNF	.1015	#38	79.6%
.1380-32UNC	.1065	#36	77.6%
.1380-40UNF	.1130	#33	77.0%
.1640-32UNC	.1285	#30	87.4%
.1640-36UNF	.1360	#29	77.6%
.1900-24UNC	.1495	#25	74.8%
.1900-32UNF	.1590	#21	76.4%
.2160-24UNC	.1770	#16	72.1%
.2160-28UNF	.1820	#14	73.3%
.2160-32UNEF	.1850	#13	76.4%
.2420-20UNC	.1930	#10	75.4%
.2420-24UNF	.2010	#7	75.7%
.2420-32UNEF	.2090	#43	81.3%

Thread:	Tap Drill:	Tap Drill:	% Thd:
.2500-20UNC	.2010	#7	75.4%
.2500-28UNF	.2130	#3	79.8%
.2500-32UNEF	.2188	7/32	77.0%
.3125-18UNC	.2570	F	76.9%
.3125-24UNF	.2720	I	74.8%
.3125-32UNEF	.2813	9/32	77.0%
.3750-16UNC	.3125	5/16	77.0%
.3750-24UNF	.3320	Q	79.4%
.3750-32UNEF	.3438	11/32	77.0%
.4375-14UNC	.3680	U	74.9%
.4375-20UNF	.3860	W	79.3%
.4375-28UNEF	.4040	Y	72.2%
.5000-13UNC	.4219	27/64	78.2%
.5000-20UNF	.4531	29/64	72.2%
.5000-28UNEF	.4688	15/32	67.4%
.5625-12UNC	.4844	31/64	72.2%
.5625-18UNF	.5000	½	86.6%
.5625-24UNEF	.5156	33/64	86.6%
.6250-11UNC	.5313	17/32	79.4%
.6250-18UNF	.5625	9/16	86.6%
.6250-24UNEF	.5781	37/64	86.6%

Thread:	Tap Drill:	Tap Drill:	% Thd:
.7500-10UNC	.6563	21/32	72.2%
.7500-16UNF	.6875	11/16	77.0%
.7500-20UNEF	.7031	45/64	72.2%
.8750-9UNC	.7656	49/64	75.8%
.8750-14UNF	.7969	51/64	84.2%
.8750-20UNEF	.8281	53/64	72.2%
1.0000-8UNC	.8750	7/8	77.0%
1.0000-12UNF	.9219	59/64	72.2%
1.0000-20UNEF	.9531	61/64	72.2%
1.1250-7UNC	.9844	63/64	75.8%
1.1250-12UNF	1.0469	1-3/64	72.2%
1.1250-18UNEF	1.0625	1-1/16	86.6%
1.2500-7UNC	1.1094	1-7/64	75.8%
1.2500-12UNF	1.1563	1-5/32	86.6%
1.2500-18UNEF	1.2031	1-13/64	65.0%
1.3750-6UNC	1.2188	1-7/32	72.2%
1.3750-12UNF	1.2969	1-19/64	72.2%
1.3750-18UNEF	1.3281	1-21/64	65.0%
1.5000-6UNC	1.3281	1-21/64	79.4%
1.5000-12UNF	1.4219	1-27/64	72.2%
1.5000-18UNEF	1.4375	1-7/16	86.6%

Formed Thread Tap Drill Chart

Thread:	Tap Drill:	Tap Drill:	% Thd:
.0600-80UNRF	.0520	#55	94.1%
.0730-64UNRC	.0635	#52	89.4%
.0730-72UNRF	.0670	#51	63.5%
.0860-56UNRC	.0760	#48	82.4%
.0860-64UNRF	.0781	5/64	74.1%
.0990-48UNRC	.0890	#43	70.6%
.0990-56UNRF	.0890	#43	82.4%
.1120-40UNRC	.0995	#39	73.6%
.1120-48UNRF	.1015	#38	74.1%
.1250-40UNRC	.1110	#34	82.4%
.1250-44UNRF	.1130	#33	77.7%
.1380-32UNRC	.1200	#31	84.7%
.1380-40UNRF	.1250	1/8	76.5%
.1640-32UNRC	.1470	#26	80.0%
.1640-36UNRF	.1495	#25	76.8%
.1900-24UNRC	.1695	#18	72.4%
.1900-32UNRF	.1730	#17	80.0%
.2160-24UNRC	.1935	#10	79.4%
.2160-28UNRF	.1960	#9	82.4%
.2160-32UNREF	.1990	#8	80.0%
.2420-20UNRC	.2130	#3	85.3%
.2420-24UNRF	.2210	#2	74.1%
.2420-32UNREF	.2280	#1	65.9%

Thread:	Tap Drill:	Tap Drill:	% Thd:
.2500-20UNRC	.2210	#2	85.3%
.2500-28UNRF	.2280	#1	90.6%
.2500-32UNREF	.2340	A	75.3%
.3125-18UNRC	.2813	9/32	82.7%
.3125-24UNRF	.2900	L	79.4%
.3125-32UNREF	.2969	19/64	73.6%
.3750-16UNRC	.3438	11/32	73.6%
.3750-24UNRF	.3480	S	95.3%
.3750-32UNREF	.3594	23/64	73.6%
.4375-14UNRC	.4040	Y	69.0%
.4375-20UNRF	.4130	Z	72.1%
.4375-28UNREF	.4219	27/64	64.4%
.5000-13UNRC	.4531	29/32	89.6%
.5000-20UNRF	.4688	15/32	91.9%
.5000-28UNREF	.4844	31/64	64.4%
.5625-12UNRC	.5156	33/64	82.7%
.5625-18UNRF	.5313	17/32	82.7%
.5625-24UNREF	.5469	35/64	55.2%
.6250-11UNRC	.5781	37/64	75.8%
.6250-18UNRF	.5938	19/32	82.7%
.6250-24UNREF	.6094	39/64	55.2%

Thread:	Tap Drill:	Tap Drill:	% Thd:
.7500-10UNRC	.6875	11/16	91.9%
.7500-16UNRF	.7188	23/32	73.6%
.7500-20UNREF	.7188	23/32	91.9%
.8750-9UNRC	.8125	13/16	82.7%
.8750-14UNRF	.8281	53/64	96.5%
.8750-20UNREF	.8438	27/32	91.9%
1.0000-8UNRC	.9375	15/16	73.6%
1.0000-12UNRF	.9531	61/64	82.7%
1.0000-20UNREF	.9688	31/32	91.9%
1.1250-7UNRC	1.0469	1-3/64	80.4%
1.1250-12UNRF	1.0781	1-5/64	82.7%
1.1250-18UNREF	1.0938	1-3/32	82.7%
1.2500-7UNRC	1.1719	1-7/64	80.4%
1.2500-12UNRF	1.2031	1-13/64	82.7%
1.2500-18UNREF	1.2188	1-7/32	82.7%
1.3750-6UNRC	1.2813	1-9/32	82.7%
1.3750-12UNRF	1.3281	1-21/64	82.7%
1.3750-18UNREF	1.3438	1-11/32	82.7%
1.5000-6UNRC	1.4063	1-13/32	82.7%
1.5000-12UNRF	1.4531	1-29/64	82.7%
1.5000-18UNREF	1.4688	1-15/32	82.7%